N87-26003

YESTERDAY, TODAY, AND TOMORROW--A PERSPECTIVE OF CFD AT NASA'S AMES RESEARCH CENTER

Paul Kutler* and Anthony R. Gross†
NASA Ames Research Center

I. ABSTRACT

This paper affords the opportunity to reflect on the computational fluid dynamics (CFD) program at NASA's Ames Research Center--its beginning, its present state, and its direction for the future. Essential elements of the research program during each period are reviewed, including people, facilities, and research problems. The burgeoning role that CFD is playing in the aerospace business is discussed, as is the necessity for validated CFD tools. The current aeronautical position of this country is assessed, as are revolutionary goals to help maintain its aeronautical supremacy in the world.

II. INTRODUCTION

The discipline of computational fluid dynamics (CFD) at NASA Ames Research Center has been a mainstay in its research program for over 15 years, and is predicted to remain so well into the next century. It is a technical discipline that was nurtured at Ames and has experienced considerable growth as a result of its demonstrated potential to aid in "building a better plane." It has received considerable support from local senior management and leaders from NASA Headquarters. The Numerical Aerodynamic Simulation (NAS) Facility is testimony to that fact.

In the following sections the past, present, and future of CFD at Ames is discussed. In each of those sections, some of the research efforts and scientists are mentioned. In addition, the facilities at their disposal for carrying out their research are discussed. In the section on Tomorrow, in addition to the planned research program, a discussion is presented on CFD validation, computer technology, and artificial intelligence.

In the Concluding Remarks section, NASA's role in fluid dynamics is discussed and an assessment on our country's current aerospace industry is made. National aeronautical goals that should challenge the aerospace engineer for many years and keep this country in a lead position are reiterated.

*Chief, Fluid Dynamics Division.
†Deputy Chief, Fluid Dynamics Division.

The authors would like to acknowledge the consultation provided by Ms. Marcie Smith regarding the history of the central computing facility and the consultation provided by Ms. Alison Andrews regarding artificial intelligence.

III. YESTERDAY'S ACCOMPLISHMENTS

Computational fluid dynamics at Ames Research Center dates back to the days when mechanical calculators were used by computationalists to arduously and meticulously obtain results to very simple (by today's standards) linear aerodynamics problems. This period occurred before CFD was a formal program at Ames and therefore will not be discussed. We would, however, like to begin the history review in the fall of 1968. At that time two Ames scientists, Harvard Lomax and Harry Bailey, were developing research tools for the analysis and application of computer software to simulate fluid flows. At the same time and with help from many others, they were assembling a hardware system for executing and visualizing the CFD software they were developing.

Their computer hardware at the time included an IBM 1800 (arriving at Ames circa 1968) linked with an IBM 2250 cathode-ray display device. Also available, but not in an interactive mode, was an IBM 7094 (circa 1962) for performing calculation in a batch mode. The beauty of the IMB 1800/2250 complex was that it was an interactive system on which the users could display their calculations while the calculations were being computed and interact with the computer to modify parameters such as the mesh size, step size, or smoothing constants. Instabilities in the numerical solution that were observed could be instantaneously ameliorated. The disadvantage of the system, of course, was its speed and storage limitations (in that regard things haven't changed much).

In the fall of 1968, two graduate students (Joseph L. Steger and Paul Kutler), from Iowa State University under a cooperative arrangement with NASA Ames, joined the ranks of the Theoretical Branch led by its Acting Branch Chief, Dr. Vernon Rossow. Mentored by Lomax and Bailey, both students studied the discipline of CFD and developed research projects to be used for their dissertations. Steger studied relaxation algorithms, their analysis, and their application to transonic flows while Kutler studied explicit algorithms and their application to supersonic flows.

The potential of CFD as an analysis and design tool was recognized by senior management, and the discipline flourished at NASA Ames. Other students, such as F. R. Bailey and W. F. Ballhaus, Jr., studied the discipline and performed pioneering work. They were followed by many others who performed original applications or developed innovative procedures in CFD. Similar student programs exist today and continue to benefit the laboratory.

The computational facilities were constantly being upgraded as a result of management's belief in the discipline. The IBM 360-67 (circa 1969) was a milestone in Ames' CFD program because it could be directly linked with the IBM 2250 to provide the computer power and additional storage needed to perform more complex flow-field calculations.

This method of operation--namely, interactive computer graphics--was an enormous time-saver, especially in the early days of CFD, when there was such a large parameter space to explore. Research that would normally take a week to do by reducing data from computer paper output took a few hours on the interactive system. However, the disadvantage of this method of operation was that the developed software was not easily portable and required major modifications before it could be disseminated to outside users.

The information generated by the CFD scientists at Ames was sometimes displayed in the form of movies not only for scientific analysis, but also for presentation at technical meetings. Movies were shot directly from the screen of the IBM 2250 by an ingenious device conjured up by Lomax. The screen was shrouded in a black cloth with the camera mounted on a tripod at one end. The room was darkened, and the computer tripped the camera lens. It worked, and it provided some of the first CFD flow-field movies. This procedure was later refined to include a fiberglass shroud, and color was added by taping different colored plastic overlays on various parts of the monochrome screen.

An ambivalent time in the lives of the CFD scientists at Ames occurred when the CDC 7600 arrived (circa 1974). On the one hand, it provided computational relief because it was a much faster machine, but on the other hand, it meant that they could no longer run interactively, and that their computer codes had to be converted to operate in a batch mode.

The arrival of the Illiac IV (circa 1972) produced a traumatic time for some CFD scientists and a challenge for others. It introduced a completely new computer architecture, i.e., parallelism. It was unique with its 64 processors operating in parallel, each with the speed of a CDC 6600, and it was the first "Class VI" machine for Ames. Input and output on the Illiac was difficult at best, with little or no debugging capabilities. Hardware failures were many. It required calculation checks to validate its own accuracy (two-thirds of the processors were used to check the other third). The Illiac IV (parts of which now reside in the Smithsonian Museum in Washington, D.C.) was finally replaced by the Cray 1-S (circa 1982). That was followed by the Cray X-MP/22 (circa 1984) and the Cray X-MP/48 (circa 1986).

In the formative stages of the CFD effort at Ames, the research program consisted of both theoretical and applied elements. The theoretical efforts involved the development and analysis of numerical algorithms for solving the fluid dynamic equations. The applied program involved the solution of those equations on practical aerodynamic problems.

Much of the early theoretical research involved the development of CFD methods and techniques that could handle difficult flow regions, such as transonic speeds, unsteady flows, shock-wave/boundary-layer interaction flows and shock interaction flows. MacCormack[1,2] developed a number of algorithms and techniques that were quickly accepted and widely used, including a simple, explicit, predictor-corrector scheme, and one of the first Navier-Stokes solution algorithms. Steger[3,4] developed relaxation procedures for solving the transonic potential equation. Beam and Warming[5,6] developed an implicit approximate factorization algorithm that has been the basis for many of the present day operational codes. Lomax,[7,8] in addition to his leadership of the CFD group, made many important, pioneering contributions to the development of CFD, including numerical algorithms and analysis procedures for evaluating newly developed algorithms. Holst[9-12] developed a shock-interaction procedure, a method to simulate boattail flows, and one of the first practical airfoil and wing transonic full-potential codes (TAIR and TWING).

When the Illiac IV arrived at the Center, with its 64 parallel processors, Rogallo[13] developed a programming language called "CFD" for this radically different machine that made programming it similar to programming in FORTRAN. In addition, he developed fast-Fourier-transform schemes for it and one of the early large eddy simulation (LES) codes.

There were many important problems that were solved using CFD on these early machines; problems that were instrumental in establishing CFD as a contributing analysis tool and problems that helped mature CFD into a demonstrated and useful technology. Two important problems that helped to establish CFD as a tool that could be useful in solving practical aerodynamic problems were the modification of the highly maneuverable aircraft technology remotely piloted research vehicle (HiMAT RPRV) and the redesign of the North American Sabre 60 wing. In both of these cases, the Bailey-Ballhaus Transonic Small-Disturbance-Steady-Flow Code[14] was used to achieve designs

that met performance requirements where conventional methods had previously been unsuccessful, and with considerable cost savings relative to those methods.

Another important CFD problem that demonstrated the usefulness of CFD and resulted in a considerable savings to the government (by eliminating the need to redesign the SPRINT antiballistic missile system) was the three-dimensional shock-on-shock problem solved by Kutler[15] using his patented "shock-capturing technique." In addition, Kutler[16-18] developed several codes and techniques that have been widely used, including inviscid codes for prediciting sonic booms about simplified configurations and flows about such supersonic configurations as the Space Shuttle, and viscous codes for treating supersonic blunt bodies. He also developed a code for analyzing the flow through an advanced propeller called the prop-fan. His code for determining the flow field about the Space Shuttle Orbiter was a major step forward in establishing CFD as a viable analysis tool.

IV. TODAY'S PROGRAM

Today the capability of CFD to effectively and accurately simulate the complexities of fluid flows of practical interest has grown tremendously relative to the capability of even a few years ago, and stands as a testament to the ingenuity and imagination of both the computer system developers and the CFD scientists. A few selected examples will demonstrate the great strides that have been made in recent years, and help illustrate the present state of the art in CFD at Ames.

In 1981, researchers recognized that the building blocks for a three-dimensional Navier-Stokes code existed and that an organized effort could produce a demonstration calculation about a complex configuration. They selected the F-16A fighter was selected as the target for a full, three-dimensional, transonic computation. Its geometry was complex and during maneuvers it would generate interesting fluid physics. A group was formed led by Holst that simultaneously addressed the geometry, grid development, and flow-solver problems necessary to simulate the flow about this configuration. The key problems were 1) how to divide the flow field into subelements that were small enough to be tractable to the existing computer system, 2) how to organize and manage the massive amount of information that would be generated by such a calculation, and 3) how to display the results in a manner usable to the CFD scientist. The Calma CAD/CAM system was used to develop the computer geometry, with data supplied by General Dynamics. Using the Ames ARC3D flow code as a basis, and the concepts of the two-dimensional GRAPE grid genration code, the flow field was divided into 16 separate zones to facilitate the initial computations on the Cray X-MP/22 computer.

Present computations are done on the Cray X-MP/48, a much larger and faster machine. A key advance was made during this program with the development of a boundary-condition procedure for the efficient transfer of information across each zonal boundary within the flow field. Significant efficiencies were introduced into the flow-solver coding, including accelerating the convergence of the basic algorithm by a factor of 40. Typical results are shown in figures 1 and 2. Figure 1 shows the grid system developed for the F-16A, and the flow-field results are shown in figure 2. Notice the great detail in the separated flow regions on the wing, resulting in a simulation that compares well with wind tunnel experiments.

The problem of simulating the flow about multiple moving bodies was a difficult one, and one whose solution provided the basis for many important applications. Computationally simulating such a flow usually involves one grid system moving relative to another, with the attendant requirement that information be passed across the interface boundary in a time-accurate basis. Rai[19,20] developed such a solution procedure that has resulted in the ability to simulate the flow through rotor-stator systems in two dimensions. The results have been spectacular in terms of visual and flow system detail. Figure 3 presents a simulation of the supersonic flow through a cascade of circular-arc airfoils, showing details of the shock and expansion wave interactions. This result was used on the cover of the March 25, 1985, issue of Aviation Week and Space Technology. In figure 4 the flow through a cascade of typical jet engine airfoils is shown, illustrating the detail to which the flow can be simulated. The use of present graphic workstation technology makes it practical to show the development of alternating trailing-edge vortices, and to show how these are propagated through the cascade system.

With the incentive to understand the fluid physics and to increase the thrust performance of the Space Shuttle Main Engine (SSME) without major modifications, thus resulting in a greater payload capability, a cooperative computational program with industry, was undertaken. This program resulted in the development of an incompressible Navier-Stokes code that could treat the complex internal geometry of the engine powerhead. Kwak,[21-24] beginning in 1981, developed a three-dimensional, incompressible Navier-Stokes code (INS3D) for that application that has provided an extensive low-speed simulation capability. Working with engineers from Rocketdyne (the builders of the SSME), researchers applied this code to simulate the flow within the hot-gas manifold, transfer ducts, and main injector of the SSME. It was determined that flow within the present three-duct design (fig. 5) was inefficient. A large, separated-flow region existed in the center duct,

and it was transmitting only 9% of the total flow. A proposed two-duct design showed significantly improved flow characteristics. As a result of this demonstration, it has been determined by Rocketdyne management that CFD simulation will be used to analyze all future SSME designs.

Using a parabolized Navier-Stokes code (PNS), Chaussee[25,26] has developed a flow-simulation capability for supersonic configurations, including the Space Shuttle Orbiter. This research program was also used to develop the first graphics application on the new Silicon Graphics IRIS workstations. The flow field about the Orbiter at reentry conditions is shown in figure 6, a picture that has appeared in many publications during the past year. The development and propagation of vortices are clearly shown, as is the flow along the surface. With present workstation capability it is possible for the computational aerodynamicist to carefully examine any selected aspect of the flow field, once a complete solution has been obtained.

Building on the PNS code technology, Rizk and Chaussee[27] extended the CFD simulation capability to hypersonic speeds and applied it to a configuration similar to that of the National Aerospace Plane (NASP). Figure 7 shows the pressure contours about a research configuration at Mach 25. Additional flow-field realism will be obtained by including real gas chemistry effects in the computations.

Modern aircraft wings, including those with tip stores, frequently show pronounced aeroelastic effects. Guruswamy and Goorgian[28-31] have combined a flow simulation code with a structural response code to develop an aeroelasticity simulation capability. It has been applied to the B-1 and F-5 aircraft wings. Typical results are shown in figure 8, where the pressure distributions for a wing with and without a tip missile are presented.

Turbulence, its formation and propagation, is not very well understood, and it is very difficult to predict for the fluid dynamicist. It is very important to be able to model the behavior of turbulence for use in Reynolds-averaged Navier-Stokes codes. With modern CFD software simulation tools in conjunction with the latest computer systems at Ames, the ability to computationally simulate turbulent flows is possible (i.e., for simple shapes and low Reynolds numbers). The program at Ames is the most advanced in the nation and possibly in the world. Both LES and direct simulation (DS) techniques are employed, with computations on three-dimensional grids taking about 100 hr of computer time. A typical result from an LES calculation by Kim and Moin[32,33] is shown in figure 9, which depicts a horseshoe vortex.

The CFD scientists at Ames have been fortunate in having easy access to some of the best computational facilities in the world. These facilities have included advanced supercomputers, graphics workstations, and a myriad of support systems, such as a VAX farm to serve as front- and back-end processors for the supercomputer solutions, and personal computers and modems to permit maximum flexibility in accessing the main computer systems.

The Central Computer Facility (CCF) provides a wide range of support for CFD scientists, from supercomputer systems to individual terminals, graphics support, and communications support. The present CCF consists of two supercomputers, a number of DEC VAX advanced minicomputers, and many smaller supporting systems. The supercomputers include a Cray X-MP/48, with four processors and 8 million 64-bit words of memory, and a CDC Cyber 205, which is a four-pipe machine with 8 million words of memory. The Cyber 205 was installed in June 1984, and has been extensively used by the turbulence physics researchers and the computational chemists. Data storage systems are an important part of the computational facility, and have been growing steadily in size and speed to support the new supercomputer systems and the new, more complex problems. For example, there is a 128-million-word solid-state disk (SSD) connected to the Cray X-MP that greatly enhances the practical size of problems that can be solved. In addition to the computational facilities, Ames research scientists benefit from a management policy that provides maximum access to the machines, without regard for individual program funding. It is a true Centerwide resource, available to everyone. The result of this policy is an open-access system, with a greater opportunity and incentive for innovation and the encouragement to experiment with new ideas.

In addition to the CCF capability, Ames has been chosen as the site of the NASA Numerical Aerodynamic Simulation (NAS) program. The NAS program is a national computing system that is designed to provide a large, fast computational system dedicated to solving aerospace problems. It will also serve as a pathfinder in the development of advanced-numerical-simulation technology and techniques. The governing concept is to remain at the leading edge of research computers by utilizing a continuing series of high-speed processors (HSP), each to be the prototype of the latest available technology, which will ensure availability for the scientists of the maximum computational capability. Through a network of satellite and land lines, researchers from all over the country can access this new capability. A new 90,000-ft^2 building has been constructed to house the NAS system and the CFD scientists who will use that capability at Ames. The first HSP, HSP-1, is the Cray 2, with a sustained operating speed of 250 Mflops, 256 million words of memory, and over a gigabyte of mass storage. It has been operational since July 1986. The second HSP is planned to be installed during 1987.

In addition to the advanced "number-crunching" capability, and the extensive communications network, computer graphics has been recognized as a critical element in the present numerical simulation process. Such tasks as developing the geometry for a complex configuration, building a multizone, three-dimensional grid system, and examining the details of a complicated flow solution, require a high-resolution, high-throughput graphics capability to most effectively exploit both the computer system and the scientists' creativity. A large number of state-of-the-art graphic workstations have been provided to the Ames scientists to both display computed results and to develop advanced graphics software. The Silicon Graphics IRIS workstation is the most common example, with upgrades being installed as they become available.

Finally, Ames utilizes a large number of ancillary systems to both support and augment the main computing systems and the work of the CFD scientists. These include both front and back-end processors to facilitate the solution procedure, as well as "smart" and "dumb" terminals to interface with the CFD scientists. A VAX farm, using both VAX 11/780-785 and VAX 8650 machines, serves as the pre- and post-data processors. Personal computers and dedicated VAX terminals serve the interface function, in addition to the IRIS graphics workstations. Finally, modems are provided to those who wish to augment their normal working time with time at home, using either their own computer, or one borrowed from their office, which permits them to connect to the Ames computer system.

V. TOMORROW'S RESEARCH PROGRAMS

NASA Ames has a vision labeled "Computation to Flight" that will govern the CFD research performed at the Center:

> "Ames will be known for its capabilities in computational analysis, experimental investigations, flight simulations and flight testing, and will be acknowledged as the lead Center in the integration of these capabilities into a technology for the design of aerospace vehicles. This integrated capability will also be used to advance basic aerodynamics science, particularly the understanding of those real fluid- and aero-dynamic phenomena that determine component and total configuration performance."

In support of that vision, Ames has defined various "targets of opportunity," i.e., a set of more refined statements of the future, expressed in terms that are measurable. These targets have been divided into three categories: 1) integrated programs, 2) aerodynamic science, and 3) research tools. Short descriptors for the targets in the three categories include: under 1) integrated programs such areas as high-angle-of-attack aerodynamics (HARV), circulation control (X-Wing), powered lift (ASTOVL), and hypersonics (NASP); under 2) aerodynamics science such areas as turbulence, viscous flows, chemically reacting flows, unsteady aerodynamics, advanced rotorcraft, space technology, and interdisciplinary physics; and under 3) research tools in such areas as algorithm enhancements, advanced computational/ experimental facilities, and advanced instrumentation.

Most of the projects under "integrated programs" satisfy the five criteria used to determine whether or not Ames embarks on a CFD research program, i.e., 1) is the problem of national importance, 2) will its solution lead to a new design tool, 3) will it aid the understanding of complex fluid physics or the discovery of new flow phenomena, 4) will it push the state of the art in computational fluid dynamics, and 5) is the problem tractable in a finite amount of time.

Three important areas that are addressed in this section and will support the planned research programs mentioned above include CFD validation, computer technology, and artificial intelligence. A perspective of each is presented below.

CFD Validation

Computational fluid dynamics is experiencing greater visibility by the aerospace community as a tool to aid in the aerospace vehicle design process. Along with its acceptance comes the requirement by the users for validation, i.e., a measure of the accuracy of the results produced by the computer code and its range of validity. Because of this understandable and justifiable requirement, CFD is beginning to play a dominant role in stimulating validation experiments and in the development of advanced instrumentation for extracting validation data.

The aerospace research community is undergoing a cultural change. In the past, computationalists and experimentalists worked somewhat autonomously. The experimenters performed their experiments to understand the fluid physics or obtain design information and compared their data with available theory, while the computationalists performed their calculations and compared their results with available experimental data, theory, or other numerical results. This process involved little or no communication between the two camps. Anonymous was heard to say, "No one believes the analysis except the engineer who performed the calculations. Everyone believes the data except the engineer who performed the tests."

What is happening today is a result of the the need for validated computer codes by the aerospace community. The two camps are now beginning to work more closely together in an attempt to get not better experimental data, but the right

experimental data to validate the code. It is important that both concerns are treated as equal partners in this endeavor. Code validation is an evolutionary process. This cultural change, as any change, takes time, but it is happening, and the results should be enhanced CFD design tools.

Experiments are generally classified as building block, benchmark, or design. Design experiments or configurational experiments involve drag, lift, moment, heat-load, and shear-load measurements. Those measurements are obtained as close to the flight conditions as possible. Benchmark or parametrical experiments obtain surface quantities, flow-field quantities at selected locations, and the tunnel boundary conditions. These data are obtained by varying the Mach number, Reynolds number, and angle of attack over the flight range. Building block or phenomenological experiments measure surface quantities; flow-field quantities; turbulence (individual stresses, correlation lengths, structure); and boundary conditions. These data are taken at representative flight Mach and Reynolds numbers. Both benchmark and building block experiments are required for CFD validation purposes.

It is important that computational tools be validated to build the confidence of their users. This is best accomplished not only by performing experiments, but also by performing numerical tests. In general, a computer code's limitations are known to the author. These limitations are based in part on the equation set solved (small disturbance, Euler, parabolized Navier-Stokes, Reynolds-averaged Navier-Stokes, full Navier-Stokes) and the accuracy of the algorithm employed (steady or unsteady, explicit or implicit, first, second, or third order). From a computational point of view, grid refinement studies can be performed to determine error-prediction estimates. Code authors should specify the range of validity of their code and export them with those restrictions. These restrictions can, of course, be removed in time as the code's range of validity increases. For example, codes should be structured to readily accept new turbulence models and this flexibility increases their potential range of validity. In general, however, codes can never be fully validated.

There are various degrees of code validation that can inform the user as to a computer code's range of applicability. Dash[34] suggests four levels of validation. They include 1) basic operationality, 2) accuracy on unit problems, 3) accuracy on component problems, and 4) accuracy in predicting overall configuration performance. In the following description of the levels of validity, "a code" can refer to a single computer code or a sequence of computer codes necessary to analyze a given configuration (e.g., a blunt-body code plus a PNS code plus an unsteady continuation code for analyzing a hypersonic vehicle).

At the level 1 validation, obvious "bugs" have been eliminated from the code, the user can run the code for various generic problems (e.g., the user is familiar with the code parameters and grid-generation routines), and various ranges of operation of the code have been established. In attaining the level 1 validation stage, comparisons with similar code solutions for generic configurations are established, comparisons with alternate code solutions to establish limiting form validity and to validate approximations (e.g., PNS sublayer approximation) are made, and checks on conservation of mass, momentum and energy are performed.

At the level 2 validation, the parameters in the code required to analyze fundamental unit problems or standard test cases have been established and the user is knowledgeable on the use of the code for the analysis of realistic configurations. In attaining this level, different unit problems have been run for different component codes (e.g., blunt body or unsteady continuation) and the turbulence models and their thermochemical parameters have been modified to agree with the data.

At the level 3 validation, the code is capable of accurately analyzing the flow fields about realistic component configurations and predicting observed trends. This level of validation establishes the code as a design tool for use in parametric or trade-off studies to answer questions related to individual component performance. In attaining this level, realistic data are employed to assess the code's performance. These data must clearly define the initial conditions, geometry, and flow parameters, and must exhibit well-defined parametric trends that can imply correct performance of codes, but not validate details. Data such as those obtained from the design experiments mentioned above are appropriate.

At the level 4 validation, the code is capable of accurately analyzing complete configurations. This level establishes a coupled system of CFD codes called "a code" in the foregoing discussion that can be used as a design tool for evaluating overall configuration performance. In attaining this level of validity, realistic overall design data (i.e., performance data) are employed that exhibit the effects of the coupled system components.

Computer Technology

The demand for supercomputers to solve CFD problems at Ames and for a variety of other applications is exponentially increasing with time. The definition of a supercomputer is somewhat nebulous. On the lighter side, some say it is a machine that beats the fastest IBM currently available or one that performs an infinite loop in just 2 minutes. Scientists say it is a machine

that is just a bit too slow to solve their most interesting problem.

A plot of the speed of computers as a function of year since 1950 can be seen in figure 10 for various scientific machines (private communication, Landshut, West Germany, July 1986). In this figure the speed component is separated into that attributable to either the components or the architecture. As can be seen in the early years, component technology was the reason for enhanced speed. Since 1970, however, newer architectures have been responsible for increased computer speed. Future computers will be designed for specific problems in which circuit boards will be replaced, depending on the application to be run.

A computer's electronic components are mostly made of silicon. It is the most intensively studied of all materials and is the heart of the computer. Researchers say that silicon chips can be improved for another decade. Beyond that, however, they look to gallium arsenide chips, which transmit electrons faster, but are harder to work with.

It is clear that the world needs much more computer power than is available today. The computer industry produces the equivalent of one human brain per year. To perform human intelligence on the computer, the most difficult of all simulations, a computer with a speed of 1 trillion operations per second is required. It is believed that capability should be available in 40 years.

Laser technology will continue to have an impact on computer technology. Researchers are working on an optical computer that uses light to process information at a much faster rate than present systems that use electrical current. Lasers will continue to be used to advance data-storage technology devices.

If one were to envision future computer systems for computational fluid dynamics based on the discipline's current and future capability and its insatiable thirst for CPU cycles, one could imagine a "farm" of supercomputers. They would be collocated for economies of scale. As a problem arose requiring a supercomputer, e.g., the NASP, it would be assigned a computer from the bank of supercomputers. For efficiency of operation, the computer would be "tuned" to the problem it is solving and the user's requests. Unused cycles, of course, would run background jobs. Once the specific project using that supercomputer had been completed or the problem had been solved, the supercomputer would be assigned to another problem.

The computer is not discipline-dependent. Unlike experimental facilities that have a range of applicability, computers are not limited to a specific Mach or Reynolds number. If a discipline such as transonics, supersonics, or hypersonics matures, the supercomputer farm facility is virtually unaffected and can be used for other simulations. This is in contrast to experimental simulation whose facilities could become idle and eventually "mothballed." The goal of CFD should not be to eliminate the need for experimental testing as was suggested in the past. The long-term goal of CFD should be simply to perform realistic flow simulations about aerospace vehicles and their components in order to predict performance. In the near term, CFD could eliminate the need for experimental testing of simple shapes and building block experiments and wind tunnel testing for aircraft at cruise conditions.

With improvements in both the software and hardware that exist today, it might be possible in the near future to actually fly an aircraft on the computer. Viscous flow codes exist today for simulating the flow about real aircraft, but they take a lot of time. The supercomputers are available, but they cannot compute the flow field fast enough to reflect changes in the aircraft's attitude and flow conditions for realistic visualization. Finally, the graphics display devices can depict the flow physics dramatically, but they must be made interactive with the supercomputer so that the user can control the aircraft's speed and behavior. With today's software and hardware technology, such a feat would be possible only for a two-dimensional airfoil and still portray a sense of realism.

Artificial Intelligence

The discipline of artificial intelligence (AI) has begun to play a role in some aspects of CFD, and promises to contribute to many additional aspects of CFD in the future.[35] AI is a branch of computer science concerned (from an engineering standpoint) with the study of how to program computers to do tasks at which humans are presently superior,[36] such as reasoning symbolically, understanding natural language, interpreting and understanding perceptual input, and applying common sense and/or expertise to problem-solving and decision-making. The automation of some CFD tasks, including geometry definition and discretization, code generation, code use to obtain a solution, and data reduction and interpretation, requires such capabilities.

Two areas within AI, computer symbolic mathematics and expert systems, are being applied to CFD problems with some success. MACSYMA, a symbolic mathematics program, has been used to analyze the stability and accuracy of numerical algorithms and to generate FORTRAN code from partial differential equations.[37,38] Expert systems, or knowledge-based systems, are AI programs which contain enough domain-specific knowledge (gleaned from experience as well as from textbooks) to enable them to perform at the level of a human expert in that domain. There are several first-generation CFD expert systems that have demonstrated the potential of this approach.[39] Among these pioneering efforts is a system that uses an

expert's heuristics to guide aerodynamic design of simple turbomachinery components,[40,41] a program which prepares the input parameters and aids in output analysis for users of the PANAIR linear aerodynamic design code,[42,43] a system embedded within an adaptive grid generator which performs the error recovery required by difficult cases,[44] and a system which partitions a two-dimensional flow field into well-shaped zones which are then individually discretized (in progress) (Andrew, A. E., unpublished work). These exploratory systems will provide the foundation for future generations of CFD expert systems. Use of the straightforward techniques will spread to many other areas within CFD, and the further development of the riskier techniques will enable more advanced applications.

One area to which application of present and future AI techniques may spread is intelligent data reduction, interpretation, and display. AI learning programs[45] may be of some use in detecting meaningful patterns in the vast amount of fluid dynamic data being generated both numerically and experimentally. Present software display packages are passive in that they display only what they are told to display, resulting in the possibility that interesting or undiscovered fluid physics might remain buried within the data. A system with knowledge of fluid dynamics and numerics might be useful in ferreting out flow phenomena (such as tertiary separation hidden in the confines of the high-resolution grid).[46]

The first step toward an experimental fluid dynamics counterpart to the intelligent data search described above has been taken in a project entitled "Smart Probe."[47] In this project, a conventional, computerized, probe-traversing mechanism has been augmented by a simple rule-based expert system to enhance its performance. The system locates regions of interesting fluid physics and homes in on those regions, probing each with increased resolution. With the same number of flow field samples, the enhanced system yields much better resolution of the flow phenomena of interest than the conventional system.

VI. CONCLUDING REMARKS

The future of fluid dynamics, both computational and experimental, is bright; it offers many challenges and should provide some exciting times. It is believed that most problems amenable for solution using validated CFD tools can be solved if the desire and resources are devoted to it. NASA's role in fluid dynamics will be to

1. Provide technology for industry when industry does not have the means.

2. Perform high-risk research that industry cannot afford to perform.

3. Perform basic computational fluid physics research to explain and discover flow phenomena.

4. Integrate the technologies of aerodynamic simulation, i.e., experimental, computational, and flight, to generate synergy that results in construction of a "better plane."

5. Work collaboratively with industry and universities to promulgate the technology.

6. As an added benefit, explore the application of CFD to other disciplines, e.g., fluids in space, medicine, hydrodynamics, automotive aerdynamics, etc.

It is important that NASA in its attempt to maintain its lead and remain a pioneer in the field of CFD phase out developed technologies and design new ones. Challenge breeds productivity in research scientists; communication speeds the process. It is management's duty to support the CFD scientist with the latest facilities, a good working environment, and the freedom to perform creative and innovative research.

The lead and preeminance that the United States enjoys in the commercial aircraft business is being challenged today by other countries. This is partially explained by the following: The intellect in fluid- and aero-dynamics in this country is not unique. This country's academic institutions and research laboratories have contributed immensely to the educational process of foreign scientists with the latest technology this country has to offer. The gap between the computing facilities of this country and the rest of the world is narrowing. The United States is no longer the sole manufacturer and owner of supercomputers. The Japanese are formidable competitors. Experimental facilities among competitive countries are on par with each other. Other countries realize the benefits of a strong aeronautical policy on their country's economics. According to Bulkeley,[48] pushing research is the only way the U.S. and other developed nations can maintain their economies because less developed nations will learn to produce less technically demanding materials and products.

It's not news that we live in an "information society." Computers are partly responsible for this glut of information. The management of this information is crucial to technological advancement in the aerospace business. With bigger and faster scientific computers on the drawing boards, validated CFD codes will generate enormous quantities of data on flows about aerospace vehicles and their components and hence play a much larger role in the design cycle of future aircraft. This country, with its CFD capability as a trump card, is in a good position to wage "technological warfare" with the rest of the world in the commercial aircraft business. It possesses all the necessary ingredients for success--intellect, facilities, freedom, and competitive spirit.

To maintain our competitive edge and preeminence, we must not continue to make evolutionary advancements; instead, we must make revolutionary jumps. Steps such as those outlined in the National Aeronautical R & D Goals[49] are appropriate. They include

1. Subsonics Goal: To build transcentury renewal that envisions technology for an entirely new generation of fuel-efficient, affordable U.S. aircraft.

2. Supersonics Goal: To attain long-distance efficiency by developing pacing technologies for sustained supersonic cruise capabilities.

3. Transatmospherics Goal: To secure future options by pursuing research toward the capability to routinely cruise and maneuver into and out of the atmosphere with takeoff and landing from conventional runways.

Pioneering new technologies is this country's strength; pursuing the ambitious goals outlined above will help to enhance our preeminent position in aeronautics.

VIII. REFERENCES

[1] MacCormack, R. W.: The Effect of Viscosity in Hypervelocity Impact Cratering. AIAA Paper No. 69-354, April 1969.

[2] MacCormack, R. W.; and Warming, R. F.: Survey of Computational Methods for Three-Dimensional Supersonic Inviscid Flow with Shocks. AGARD-LS-64, 1973.

[3] Steger, J. L.; and Lomax, H.: Transonic Flow About Two-Dimensional Airfoils by Relaxation Methods. AIAA Journal, Vol. 10, No. 1, Jan. 1972, pp. 49-54.

[4] Steger, J. L.; and Lomax, H.: Generalized Relaxation Methods Applied To Problems in Transonic Flow. Proceedings of the Second International Conference on Numerical Methods in Fluid Dynamics, Vol. 8, Springer-Verlag, New York, 1971, pp. 193-198.

[5] Beam, R. M.; and Warming, R. F.: An Implicit Factored Scheme For The Compressible Navier-Stokes Equations. AIAA Journal, Vol. 16, No. 4, April 1978, pp. 393-402.

[6] Warming, R. F.; and Beam, R. M.: On the Construction and Application of Implicit Factored Schemes for Conservation Laws. Symposium on Computational Fluid Dynamics, New York, April 16-17, 1977, SIAM-AMS Proceedings, Vol. 11, 1978, pp. 85-129.

[7] Lomax, Harvard: An Operational Unification of Finite Difference Methods for the Numerical Integration of Ordinary Differential Equations. NASA TR R-262, 1967.

[8] Lomax, Harvard; and Inouye, Mamoru: Numerical Analysis of Flow Properties About Blunt Bodies Moving at Supersonic Speeds in an Equilibrium Gas. NASA TR R-204, 1964.

[9] Tannehill, J. C.; Holst, T. L.; and Rakich, J. V.: Numerical Computation of Two-Dimensional Viscous Blunt Body Flows with an Impinging Shock. AIAA Paper No. 75-154, Jan. 1975; Also AIAA J., Vol. 14, Feb. 1976, pp. 204-211.

[10] Holst, T. L.: Numerical Solution of Axisymmetric Boattail Flow Fields with Plume Simulators. AIAA Paper No. 77-224,, Presented at the 15th Aerospace Sciences Meeting, Los Angeles, Calif., January 24-26, 1977.

[11] Holst, T. L.: An Implicit Algorithm for the Conservative Transonic Full-Potential Equation Using an Arbitrary Mesh. AIAA Paper No. 78-1113, July 10-12, 1978; Also AIAA J. Vol. 20, Oct. 1979, pp. 1038-1045.

[12] Holst, T. L.; and Thomas, S.: Numerical Solution of Transonic Wing Flow Fields. AIAA Paper No. 82-0105, Jan. 11-14, 1982; Also AIAA J., Vol. 21, No. 6, June 1983, pp. 863-870.

[13] Hord, R. Michael: The Illiac IV, the First Supercomputer. 1982, Library of Congress No. IFBN 0-914894-71-4, Computer Science Press.

[14] Bailey, Frank R.; and Ballhaus, William F.: Relaxation Methods for Transonic Flow About Wing-Cylinder Combinations and Lifting Swept Wings, Vol. II. Problems of Fluid Mechanics, of Lecture Notes in Physics, No. 19, Proceedings of the Third International Conference on Numerical Methods in Fluid Mechanics; H. Cabannes and R. Teman (ed.), Springer-Verlag, 1973, pp. 2-9.

[15] Kutler, P.; and Sakell, L.: Three-Dimensional, Shock-on-Shock Interaction Problem. AIAA J., Vol. 13, No. 10, pp. 1360-1367, Oct. 1975.

[16] Kutler, P.; and Lomax, H.: Shock-Capturing, Finite-Difference Approach to Supersonic Flows. J. Spacecraft and Rockets, Vol. 8, No. 12, pp. 1175-1182, Dec. 1971.

[17] Kutler, P.; Warming, R. F.; and Lomax, H.: Computation of Space Shuttle FLow Fields Using Noncentered Finite-Difference Schemes. AIAA J., Vol. 11, No. 2, Feb. 1973, pp. 196-204.

[18] Kutler, P.; Pedelty, J. A.; and Pulliam, T. H.: Supersonic Flow Over Three-Dimensional Ablated Nosetips Using an Unsteady Implicit Numerical Procedure. AIAA Paper No. 80-63, AIAA 18th Aerospace Sciences Meeting, Pasadena, CA, Jan. 1980.

[19] Rai, M. M.: A Relaxation Approach to Patched-Grid Calculations with the Euler Equations. Journal of Computational Physics, Vol. 66, No. 1, Sept. 1986, pp. 99-131.

[20] Rai, M. M.: Navier-Stokes Simulations of Rotor-Stator Interaction Using Patched and Overlaid Grids. AIAA Paper No. 85-1519, July 15-17, 1985.

[21] Kwak, D.; Chang, J. L. C.; Shanks, S. P.; and Chakravarthy, S.: A Three-Dimensional Incompressible Navier-Stokes Flow Solver Using Primitive Variables. AIAA J, Vol. 24, No. 3, Mar. 1986, pp. 390-396.

[22] Chang, J. L. C.; Kwak, D.; Dao, S. C.; and Rosen, R.: A Three-Dimensional Incompressible Flow Simulation Method and Its Application to the Space Shuttle Main Engine, Part I - Laminar Flow. AIAA Paper 85-0175, Jan. 1985.

[23] Chang, J. L. C.; Kwak, D.; Dao, S. C.; and Rosen, R.: A Three-Dimensional Incompressible Flow Simulation Method and Its Application to the Space Shuttle Main Engine, Part II - Turbulent Flow. AIAA 85-1670, July 1985.

[24] Yang, R-J; Chang, J. L. C.; and Kwak, D.: A Navier-Stokes Simulation of the Spce Shuttle Main Engine Hot Gas Manifold. AIAA 87-0368, Jan. 1987.

[25] Chaussee, D. S.; Rizk, Y. M.; and Buning, P. G.: Viscous Computation of a Space Shuttle Flow Field. Ninth International Conference on Numerical Methods in Fluid Dynamics, Saclay, France, June 1984.

[26] Chaussee, D. S.: High Speed Viscous Flow Calculations About Complex Configurations. Paper 29, 58th Meeting of the Fluid Dynamics Panel Symposium on Applications of Computational Fluid Dynamics in Aeronautics, April 7-10, 1986, Aix-en-Provence, France.

[27] Rizk, Y.; Chaussee, D.; and Steger, J.: Numerical Simulation of the Hypersonic Flows Around Lifting Vehicles. Symposium on Aerodynamics of Hypersonic Lifting Vehicles Sponsored by the AGARD Gluid Dynamics Panel to be held in Bristol, United Kingdon, April 6-9, 1987.

[28] Guruswamy, G. P.; Goorjian, P. M.; and Tu. E. L.: Transonic Aeroelasticity of Wings with Tip Stores. AIAA Paper No. 86-1007-CP, May 1986.

[29] Guruswamy, P.; Goorjian, P. M.; Ide, H.; and Miller, G.D.: Transonic Aeroelastic Analysis of the B-1 Wing. Journal of Aircraft, Vol. 23, No. 7, July 1986, pp. 547-553.

[30] Guruswamy, G. P.; Goorjian, P. M.; and Tu. E. L.: Unsteady Transonics of a Wing with Tip Store. Journal of Aircraft, Vol. 23, No. 8, August 86, pp. 662-668.

[31] Guruswamy, P.; and Goorjian, P.M.: Efficient Algorithm for Unsteady Transonic Aerodynamic Aerodynamics of Low-Aspect-Ratio Wings. Journal of Aircraft, Vol. 22, No. 3, March 1985, pp. 193-199.

[32] Moin, P.; and Kim, J.: The Structure of the Vorticity Field in Turbulent Channel Flow. Part I. Analyses of Instantaneous Fields Statistical Correlations. J. of Fluid Mechanics, No. 155, pp. 441-464, 1985.

[33] Kim, J.; and Moin, P.: The Structure of the Vorticity Field in Turbulent Channel Flow. Part II. Study of Ensemble--Averaged Fields. J. of Fluid Mechanics, No. 162, Jan. 1986, pp. 339-363.

[34] Dash, S.: Code integration - Forebody/inlet/Com./Nozzle. Second National Aerospace Plane Symposium, Laurel, Maryland, Nov. 1986.

[35] Kutler, P.; Mehta, U. B.; and Andrews, A.: Potential Application of Artificial Intelligence Concepts to Numerical Aerodynamcic Simulation. NASA TM 85967, 1984.

[36] Rich, E.: Artificial Intelligence. McGraw-Hill Book Co., Inc., 1983.

[37] Roache, P. J.; and Steinberg, S.: Symbolic Manipulation and Computational Fluid Dynamics. AIAA Paper 83-1952, July 1983.

[38] Steinberg, S.; and Roache, P. J.: A Toolkit of Symbol Manipulation Programs for Variational Grid Generation. AIAA Paper 86-0241, Jan. 1986.

[39] Andrews, A. E.: Progress and Challenges in the Application of Artificial Intelligence to Computational Fluid Dynamics. AIAA Paper 87-0593, Jan. 1987.

[40] Tong, S. S.: Design of Aerodynamic Bodies Using Artificial Intelligence/Expert System Technique. AIAA Paper 85-0112, Jan. 1985.

[41] Tong, S. S.: Coupling Artificial Intelligence and Numerical Computation for Engineering Design. AIAA Paper 86-0242, Jan. 1986.

[42] Conner, R. S.; Purdon, D. J.; and Wamsley, F. K.: Pan Air Consultation Using Expert System Techniques. AIAA Paper 85-4094, Oct. 1985.

[43] Conner, R. S.; and Purdon, D. J.: PAN AIR Knowledge System. AIAA Paper 86-0239, Jan. 1986.

[44] Dannenhoffer, J. F., III; and Baron, J. R.: Robust Grid Adaptation for Complex Transonic Flows. AIAA Paper 86-0495, Jan. 1986.

[45] Cheeseman, P.: Learning of Expert Systems From Data. Proceedings of the IEEE Workshop on Principles of Knowledge-Based Systems, Dec. 3-4, 1984, Denver, Colorado, pp. 115-122.

[46] Kutler, P.: A Perspective of Computational Fluid Dynamics. NASA TM 88246, 1986.

[47] Zilliac, G.; Cantwell, E.; and Fukunishi, Y.: Smart Probe Progress Report. Fluid Dynamics Research Branch, NASA Ames Research Center, Oct. 1986.

[48] Bulkeley, W. M.: Frontiers of Science. The Wall Street Journal, Nov. 10, 1986.

[49] Aeronautical Policy Review Committee: National Aeronautical R&D Goals. Executive Office of the President, Office of Science and Technology Policy, March 1985.

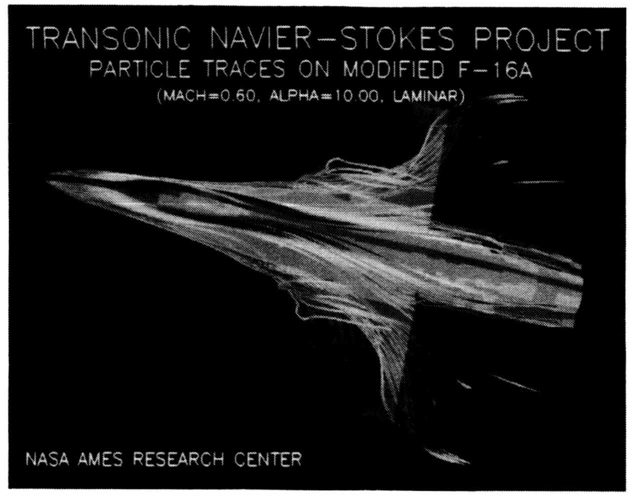

Figure 2: Particle Traces on the Modified F-16A.

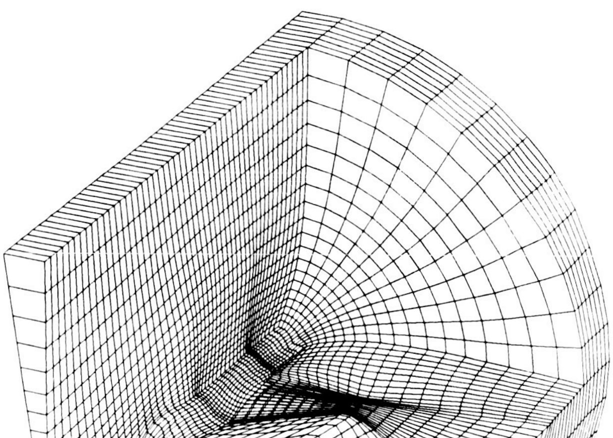

Figure 1: Grid System for the Modified F-16A Simulation.

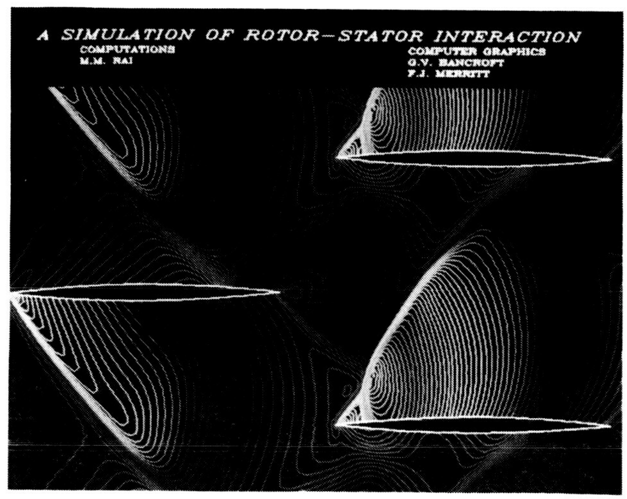

Figure 3: Supersonic Flow Through a Two-Dimensional Rotor-Stator.

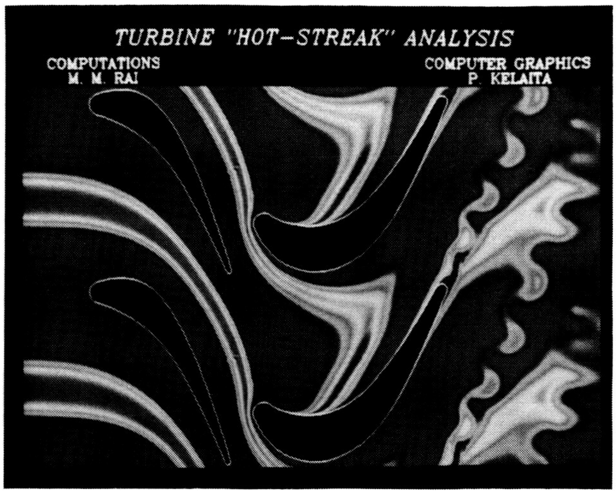

Figure 4: A Navier-Stokes Simulation of Rotor-Stator Interaction.

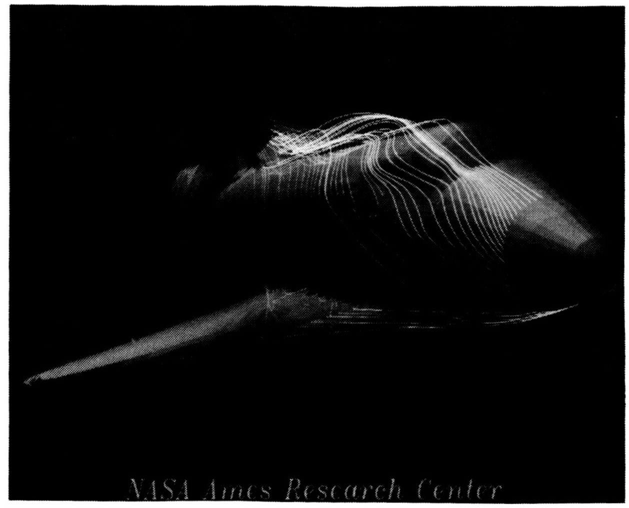

Figure 6: Flow Field About the Space Shuttle Orbiter.

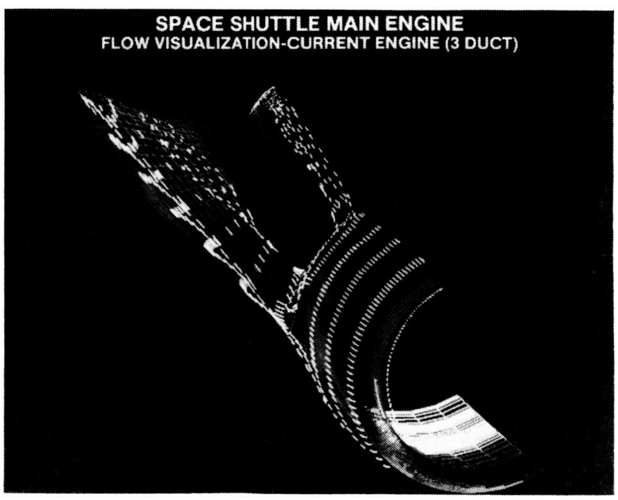

Figure 5: Flow Through Space Shuttle Main Engine Transfer Ducts.

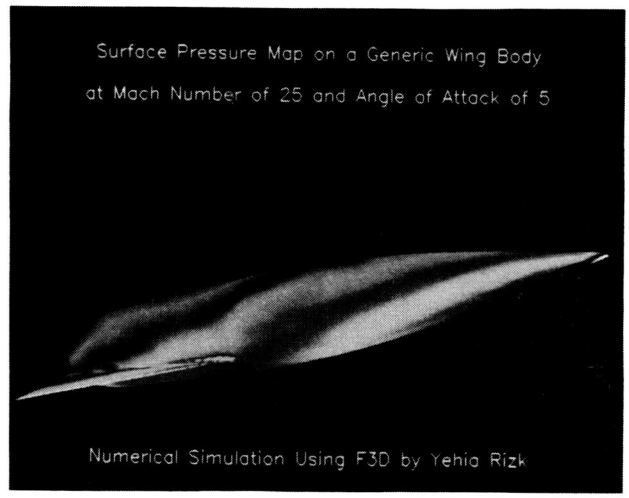

Figure 7: Pressure Contour About NASP-like Configuration.

ORIGINAL PAGE IS
OF POOR QUALITY

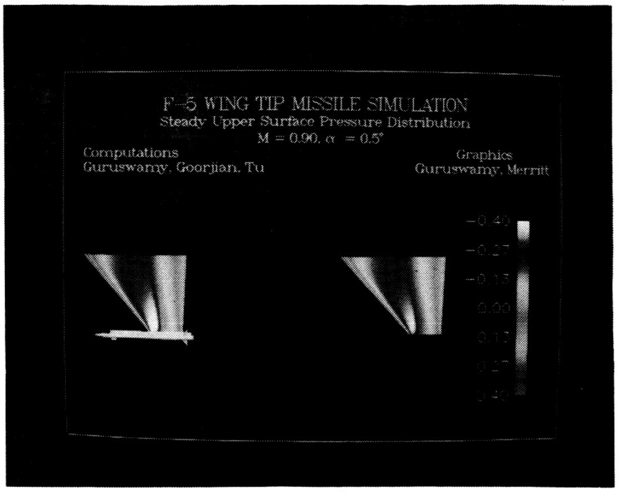

Figure 8: F-5 Wingtip Missile Simulation.

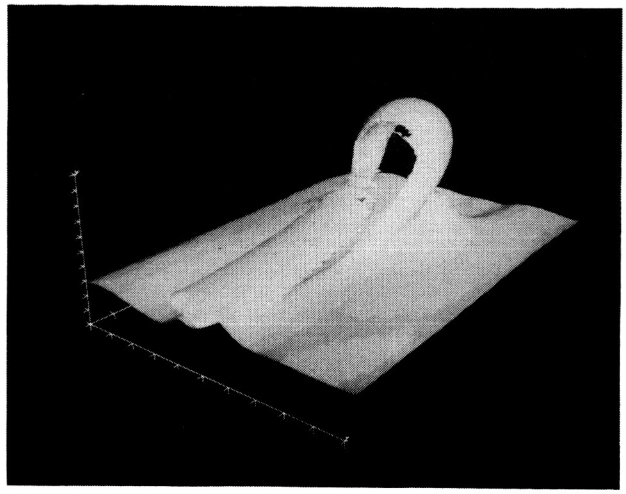

Figure 9: Horshoe Vortex Simulation.

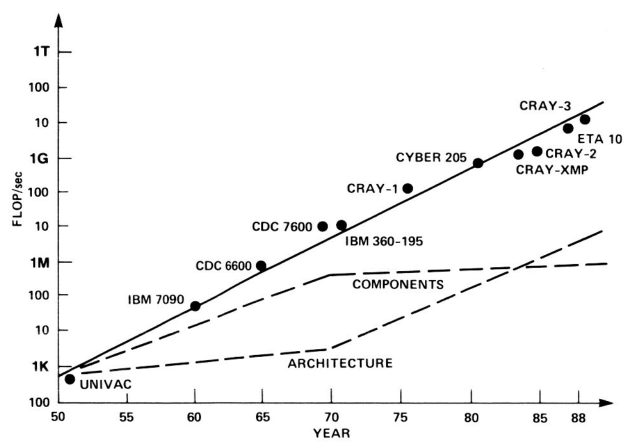

Figure 10: Computer Development Since 1950.